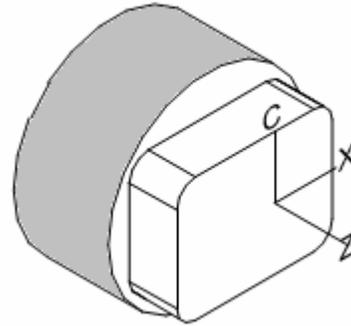


## Polar coordinate interpolation function G12.1

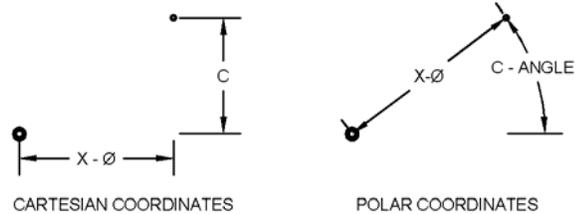
On a Turning Center that is equipped with a rotary axis (C-axis), interpolation between the linear axis “X” and the rotary axis “C” is possible by use of the G12.1-function. This function simplifies programming of shapes to be machined on the front face of a part, such as the rectangular shape with rounded corners as shown here.

Machining of such shapes is accomplished by use of an end mill that is attached to a “Z-axis live tool attachment” with the end mill pointing toward the front face of the part.

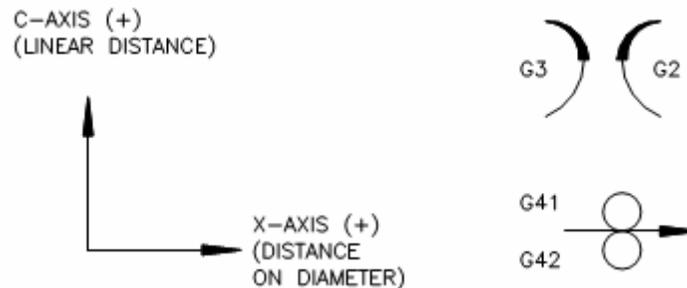


Programming with the G12.1-function is done on the X-C coordinate system plane. In this coordinate system plane the C-axis is regarded as a linear axis instead of a virtual rotary axis.

Programming is done similar to the way it is done on a basic X-Y plane. Linear or circular interpolation can be done. Cartesian coordinates are used for defining either the part shape geometry or the tool path geometry. In the G12.1-mode the control converts Cartesian coordinates to Polar coordinates, automatically.



### Layout of the X-C coordinate system plane



- The diagram above shows the X-C coordinate system plane as viewed when looking at the front face of the main spindle.
- The address “X” defines a point by the distance from origin horizontally on diameter (Positive or negative value). “On diameter” means: twice the actual distance from origin.
- The address “C” defines a point by the actual distance vertically from origin (Positive or negative value). “C” is defined in linear units of measurement, not in angular units.

## ***Notes on programming with the G12.1 function***

- **Plane selection**  
The G18-plane select command must be active in G12.1-mode. On turning centers the X-Z coordinate system plane (G18) is set as the default plane. System parameter settings related to the G12.1-function are normally set to allow polar coordinate interpolation in the G18-plane.
- **Coordinate system origin**  
The Origin of the X-C coordinate system is fixed at the center of the revolving work spindle. The origin (X0, C0) must not be shifted.
- **Angular orientation of the X-C coordinate system plane**  
Angular orientation is set by the absolute C-axis angle that exists at the time when entering the G12.1 – mode. For example: when G0 C60.0 has been commanded before entering the G12.1-mode, the X-C coordinate system plane is set on a 60° angle relative to both the horizontal and vertical axis.
- **Positioning command**  
“G0” cannot be used in G12.1-mode. Positioning is done in G1- mode, using a feed rate of around 30” to 60” per minute, depending on application.
- **Feed command**  
In the G12.1-mode the feed velocity can be specified either by units of linear distance per minute (G98-mode) or by units of linear distance per spindle revolution (G99-mode). Use of excessive feed rate can adversely influence the accuracy of a machined shape. Recommend range of feed rates for polar coordinate interpolation is from 1” to around 10” per minute, depending on application. Feed rate must be reduced in case when circular interpolation is done near the X-C zero point. Velocity of the rotary axis may become excessive and as a result, servo errors or servo overload may occur.
- **Incremental axis move command**  
Address “U” can be used for incremental move command along the X-axis. U= horizontal distance from a current point to the next point – on diameter.  
Address “H” can be used for incremental move command along the C-axis. H= vertical distance from a current point to the next point.
- **Linear interpolation command**  
G1 X\_\_C\_\_F\_\_ (absolute) or: U\_\_H\_\_ F\_\_ (incremental)

Interpolation between X and Z or between C and Z cannot be done. Z-axis move command must be specified in a separate block, not together with X-axis or C-axis commands.

- **Circular Interpolation command**

(G2 or G3) X\_\_C\_\_R\_\_F\_\_ (absolute) or: U\_\_H\_\_R\_\_F\_\_ (incremental)  
Addresses X and C define the end point of an arc. The address "R" defines the radius of an arc when the included angle of the arc segment is 180° or less.

Addresses "I" and "J" can be used for defining the arc center.

Address "I" specifies the actual distance and direction (+/-) from the start point of the arc to the arc center along the X-axis.

Address "J" specifies the actual distance and direction (+/-) from the start point of the arc to the arc center along the C-axis.

Command for arc of less than 360°: (G2 or G3) X\_ C\_ I\_ J\_

Command used for full circle: (G2 or G3) I\_ or: J\_

- **Cutter compensation function**

In polar coordinate interpolation the cutter compensation function should always be used, regardless of programming method. Size control on a machined shape is done by use of the cutter compensation function, not by changing the X-offset data.

G40 must be active at the time when entering the G12.1-mode.

G41 or G42 must be commanded after the G12.1- command.

G40 should be commanded before canceling the G12.1-mode.

Cutter compensation commands should be done together with a G1-command that moves the tool onto the part or away from it. For example:

G1 G41 X\_ or C\_ F\_ ("Ramp-ON")

G1 G40 X\_ or C\_ F\_ ("Ramp-OFF")

When ramping ON or OFF along the X-axis the moving distance must be greater than or equal to twice the "R-data". When ramping ON or OFF along the C-axis the moving distance must be greater than or equal to the "R-data".

- **Cutter compensation data setting**

Cutter compensation data (R-offset) must be set under column "R" located in the tool offset data tables as follows:

a) When the program-coordinates represent the geometry of the tool center-path, the R-data is set at zero, initially.

b) When the program-coordinates represent the geometry of the actual part shape, the actual cutter radius must be input on the R-data.

The "Tool nose type"- data (located at the column "T" at the tool-offset tables) is also part of the cutter compensation data. On the offset number that is used for a milling cutter the "T" data must be set = 0.

All offset data, including the "R" and "T"-data are activated by the tool offset command. (The "D"-command, such as used in machining center programming cannot be used).

- **Adjusting the part size**

Suppose that an external hexagon shape was machined over-size by 0.005" across the flats. When the cutter compensation function has been properly applied in the program, size adjustment on this part can be done by reducing the "R"-data in the tool offset table by the amount of -0.005". Size control cannot be accomplished by adjusting the X-axis offset.

- **Cutting tools / Tool offset data**

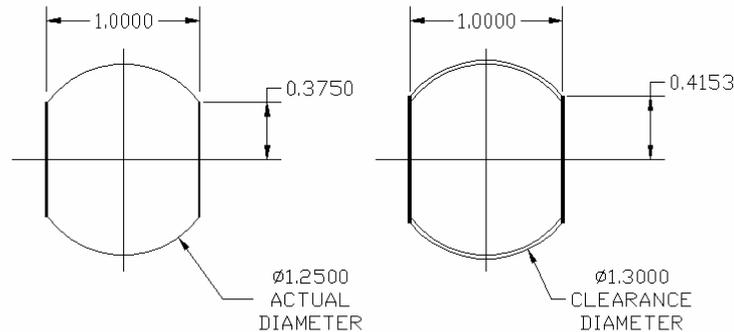
Machining in the G12.1-mode is normally done by use of a flat bottom end mill that is pointing toward the end-face of the part. This tool must be touched off (along the X-axis) at the cutter center, not at the periphery of the tool. Once the X-axis tool-offset for a given tool has been established accurately, it must not be modified later in attempting to control the size of the machined part. (Please refer to paragraph above). Erroneous tool offset data causes faulty part geometry.

In polar coordinate interpolation, a flat bottom end mill that is pointing toward the OD of the part cannot produce an acceptable part shape. On some applications, a ball-nose end mill that is pointing toward the OD of the part can be used. This type of tool must be touched off along the X-axis at the center of the ball-nose.

- **Programming the Tool approach point**

Caution must be used when positioning the milling cutter near the OD of a part. The X-axis tool offset data is based on the cutter center, not the periphery. The tool approach point for the X-axis is calculated as follows:  
 $X = \text{Part OD} + \text{Cutter diameter} + \text{clearance}.$

## Programming example



The figure above left shows two flat surfaces to be machined on the front face of a 1.25" diameter part. A clearance diameter of 1.300" that intersects with both of the flats has been added to the figure on the right. The intersections (X 1.0 C0.4153 and X1.0 C-0.4153) define start and end points in the machining program. The axial depth of the flats is assumed to be at Z-0.375" from the front face. A 1/2" diameter end mill is used for machining the two flats.

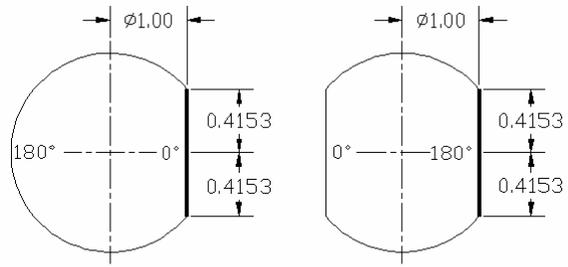
### Machining concept

- In the layout as shown in the sketch above the cutter approaches the part from the 3 o'clock position. The C-axis angle zero point is located at the 3 o'clock as well. Angles (plus direction) run counter clockwise. For machining the shape as shown above the C-axis should be positioned at zero degrees first before the G12.1 command.
- When machining the flat surfaces as in the above example, the cutter does not actually move in vertical direction. In the G12.1-mode, machining of most types of shapes is accomplished by moving the cutter along the X-axis horizontally and by rotating the C-axis at the same time.
- When X-axis coordinates are commanded with a negative sign the cutter center will not travel past X0 to the negative side of the quadrant. Instead, the C-axis is automatically rotated around so that machining on the negative quadrant can be accomplished while the cutter remains on the positive quadrant (or plus-side of the X-axis).

### Deciding the machining method

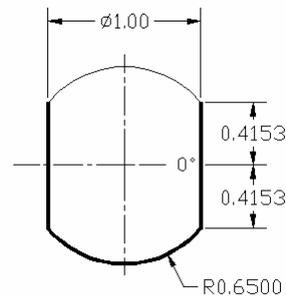
The milling operation on this part can be programmed in various different ways. For simplicity, two different programming methods A and B are shown here:

A) By this method, one of the surfaces is machined at 0-degree angle first as shown on the left figure. Next, the part is rotated 180° then the other surface is machined as shown on the figure on right.



METHOD "A"

B) By this method, both surfaces are being machined in one continuous path as shown on the figure on right. No actual machining is done on the 0.650"-radius. The cutter will clear the 1.25" part diameter. Hence the cutter can be moved around the arc at a high feed rate.



METHOD "B"

Preparing the machining program

"Climb Cutting" is done in both cases. Hence the cutting start point coordinates in both cases are at X1.0 C0.4153. The automatic cutter compensation function G41 is applied. The cutting start point is located on the top right corner on each of the figures shown. The part dimensions as shown on the sketches can be 'plugged' directly into the program.

<b>Cutting Method "A"</b>	<b>Cutting Method "B"</b>
<i>NC Program – doing each flat separately (using 1/2"-diameter cutter)</i>	<i>NC Program –doing both of the flats in one continuous path (using 1/2"-diameter cutter)</i>
N100 (MILL TWO FLATS) G40 G13.1 T0101 M5 M35 G28 H0 G97 S1000 M33 G0 Z.1 C0 X1.8 G12.1 G1 G98 C0.4153 F20. G1 G41 X1.0 F5.	N100 (MILL TWO FLATS) G40 G13.1 T0101 M5 M35 G28 H0 G97 S1000 M33 G0 Z.1 C0 X1.8 G12.1 G1 G98 C0.4153 F20. G1 G41 X1.0 F5.

<p>Z - .375 (depth of cut)  C-0.4153  G40 X1.8 F20.  G13.1  G0 Z0.1  G0 C180.  G12.1  G1 G98 C0.4153 F20.  G1 G41 X1.0 F5.  Z - .375 (depth of cut)  C-0.4153  G40 X1.8 F20.  G13.1  G0 Z0.1  G0 X—Z—M35  M1</p>	<p>Z - .375 (depth of cut)  C-0.4153  G2 X-1.0 R.650 F60.  G1 C0.4153 F5.  G40 X-1.8  G13.1  G0 Z0.1  G0 X—Z—M35  M1</p>
--	--

## SAMPLE PROGRAM

M5	Stop the main spindle
M35	C-axis select command
G40	Cutter comp cancel command
G13.1	Polar coordinate interpolation cancel command
G30 U0 W0	Tool exchange point
G28 H0	Zero return C-axis
T0808 (3/4" DIA. CUTTER)	When the cutter center path is programmed the R-offset for the tool is set = 0, initially. After inspecting the first part, adjust the R either plus or minus, as needed for size control. When
G97 S2000 M33	Live tool spindle ON
G0Z.1 C0 M8	Z-approach, C-axis at zero degrees
X2.15	X-approach ( $1.3+0.75+0.1=2.15$ )
G12.1	Polar coordinate interpolation ON
G1 G98 C.5339 F60.	C-axis position at the first point of the contour shape, use IPM - feed mode, if desired (Note: the C-command at this time represents a linear dimension – not degrees)
G1 G41 X1.75F7. (1)	Cutter comp ON X-axis position at the first point of the contour shape
Z-.88	Move the Z-axis to the desired depth on the part
G1 C-.5339 (2)	C-axis position at the second point of contour
G2 X-1.75 R1.025 F60. (3)	X-axis position at the third point of the contour (No cutting is done on the arc, a high feed rate is used for the arc move )
G1 C.5339 F7. (4)	Fourth point of the contour
G1 G40 X-2.15 F60.	Cutter comp OFF The X-axis must move at least two times the "R"-value that is used in the tool offset
G13.1	Polar coordinate interpolation OFF
G99 G0 X2.5	Retract X-axis & switch back to IPR- feed mode
Z.1 M35	Retract Z-axis and stop milling spindle
G30 U0 W0. M9	Second reference point return
M1	Optional stop

### Notes for programming in the G12.1-mode

- Program command for linear Interpolation: **G1 X\_\_C\_\_F\_\_**
- Program command for circular Interpolation: **G2 (G3) X\_\_C\_\_R\_\_F.** Letter **I\_\_** and or **J\_\_,** can be used instead of "R for arc center specification, if desired.

- Feed rate command is specified either in units of inches per minute (G98) or in units of inches per revolution (G99).
- Positioning command "G0" is not allowed while in "G12.1-mode". Positioning at speeds similar to rapid traverse is done in G1, G2 or G3-mode, using a high feed rate command
- Interpolation between X and Z-axis or between C and Z-axis is not allowed while in G12.1-mode. Z-axis moves must be commanded separately, without X or C commands.
- Cutter compensation function in G12.1-mode is available. The commands: G40, G41 and G42) must be specified while in the "G12.1-mode", not before or after.
- Cutter compensation data is activated by the tool-offset command. Cutter compensation number (such as "D --") must not be commanded.
- Cutter radius information is entered under the "R"- data at the tool-offset register. Tool-vector "T" is always set at zero.

